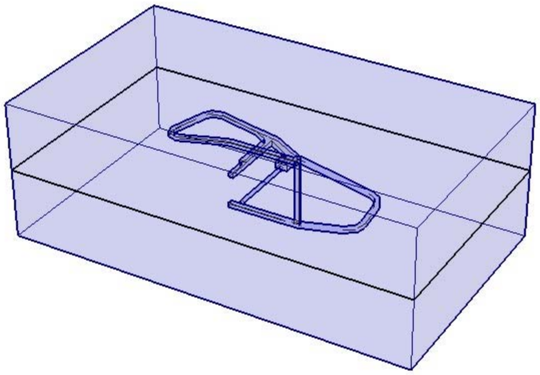
Introduction: RS^3 Tutorial 8 Underground Tunnels

Welcome to RS^3. This tutorial demonstrates how to import an Examine3D file and refine the mesh. The model is of underground tunnels under gravity stresses.



The finished product of this tutorial can be found in the **Tutorial 08 Underground Tunnels.rs3dmodel** file. All tutorial files installed with RS3 can be accessed by selecting File → Recent → Tutorials folder from the RS3 main menu.

CONTENTS

Tutorial Key Concepts
Embankment consolidation
Boreholes
Modified Cam Clay
Transient groundwater

Step 1: Starting the Model

CREATING A BLANK DOCUMENT

New Project

Start RS3 by selecting Programs → Rocscience → RS3 2.0 → RS3 from the Windows start menu. RS3 opens to a blank screen, which allows you to create a new model by pressing the [New Project] button. If the RS3 application window is not already maximized, maximize it now so the full screen space is available for use.

Save

First, save as **Tutorial 08 Underground Tunnels.rs3dmodel**: File → Save.

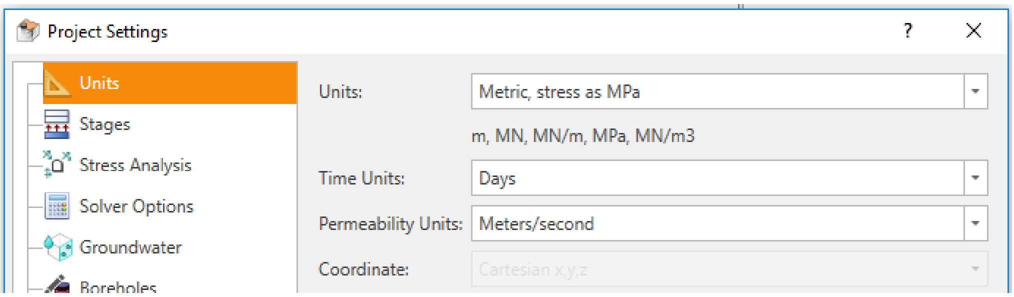
Step 2: Editing the Project Settings

CONFIGURING THE UNITS

Project Settings

The Project Settings dialog is used to configure the main analysis parameters for your RS3 model. Open the dialog through File → Project Settings. This will open the dialog on the first tab: [Units], and set Units = Metric, stress as MPa.

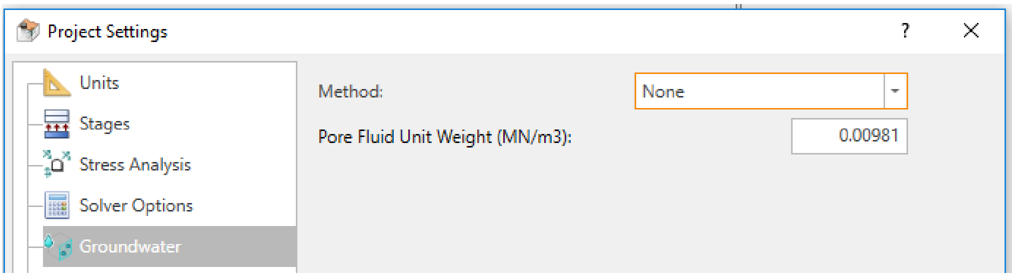
Project Settings: Units



OTHER OPTIONS

Next, select the [Groundwater] tab.

Project Settings: Ground water



Enter Method = None and select [OK] to close the dialog. Go to File → Project Summary and enter Underground Tunnels as the Project Title.

Do not change any other settings. Select [OK] to close the dialog.

Step 3: Importing Examine3D

IMPORTING THE TUNNEL GEOMETRY

Select: File → Import → Import Examine3D file.

Select **UndergroundTunnels.ex3** (from: C:\Users\Public\Documents\Rocscience\RS3 2.0 Examples\Tutorials\Tutorial 08 Underground Tunnels), ensure that when selected in the visibility pane, that in the properties pane for the underground tunnels, the Role = Excavation, and Applied Property = No Material.

Geology

Step 4: Defining the External Box

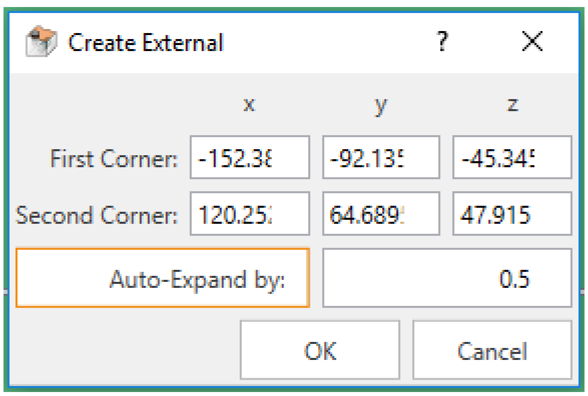
CREATING THE EXTERNAL BOX

Ensure the Geology tab is selected from the workflow at the top of the screen.

Select: Geometry → Create External Box.

A Create External dialog will open, Auto-Expand by = 0.5, press [Auto-Expand by] to update the selection, then [OK].

Create External



Geology

Step 5: Defining the Materials

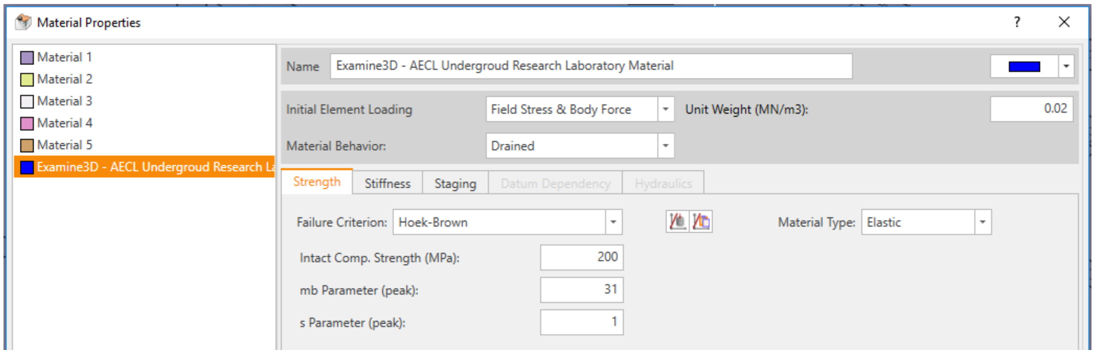
DEFINING MATERIALS PROPERTIES

Define Materials

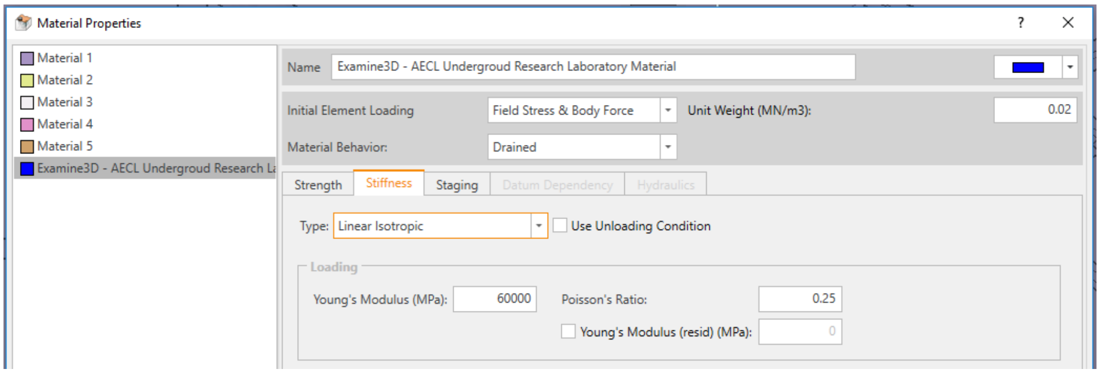
Under the same tab (Geology) you can assign the materials and properties of our model through Materials → Define Materials.

The import should have brought the material properties from the Examine3D model, select [Examine3D – AECL Underground Research Laboratory Material], and ensure the following properties are entered in the [Strength] and [Stiffness] tabs respectively

Examine3D Material Properties: Strength



Examine3D Material Properties: Stiffness



Step 6: Finalizing the Geometry

Geology

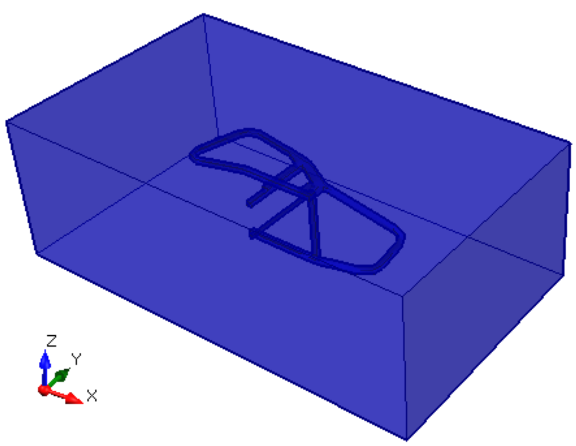
DIVIDING ALL GEOMETRY

Divide All Geometry

Now we can cut into the external box with the tunnels: Geometry → 3D Boolean → Divide All Geometry.

Select the external box in the visibility pane, and in the properties pane change the Role = Geology, Applied Property = Examine3D AECL Underground Research Laboratory Material. Your model should now appear as below:

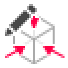
Current State of Model



Step 7: Adding Stress Loading

Loads

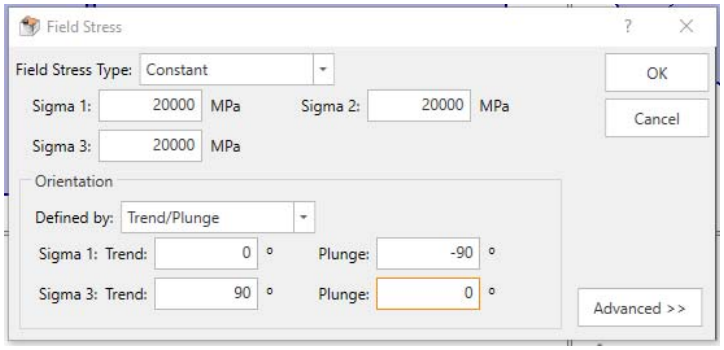
APPLYING FIELD STRESS TO THE MODEL



Field Stress

Next we go to the Loads tab. This tab allows you to edit the loading conditions. Select: Loading → Field Stress.

Field Stress



Enter Field Stress Type = Constant, Sigma 1, 2, and 3 = 20000, Defined by = Trend & Plunge, Sigma 1 (Trend, Plunge) = (0, -90), Sigma 3 (Trend, Plunge) = (90, 0), and select [OK].

Step 8: Setting Boundary Conditions

Restraints

ADDING MODEL RESTRAINTS

Auto Restrain (Underground)

Move to the Restraints tab to assign restraints to the external boundary of the model.

RS3 has a built in "Auto Restrain" tool for use on underground models. Select: Restraints → Auto Restrain (Underground).

This completes the construction of the model (in terms of geometry).

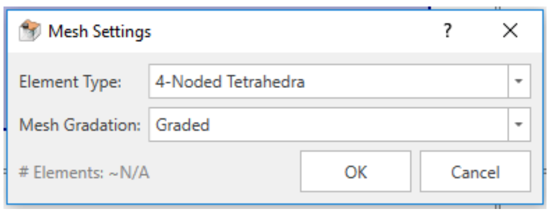
Step 9: Meshing

Mesh

CONFIGURING AND CALCULATING MESH

Mesh Settings

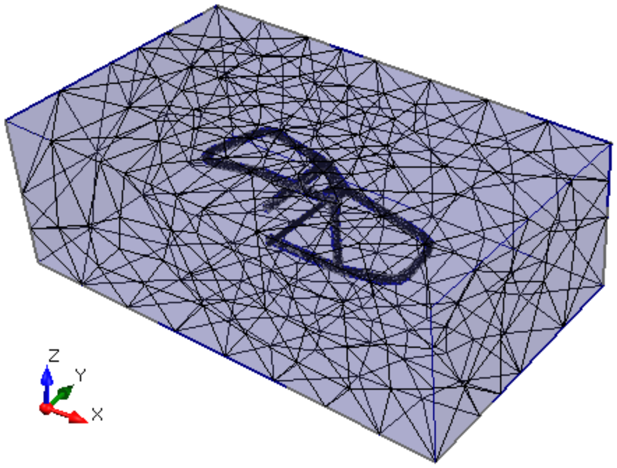
Next we move to the Mesh tab: Mesh → Mesh Settings.



Enter Element Type = 4-Noded Tetrahedra, Mesh Gradation = Graded, [OK]. Then mesh the model: Mesh → Mesh. The model should appear as below:

Mesh

Model with completed mesh



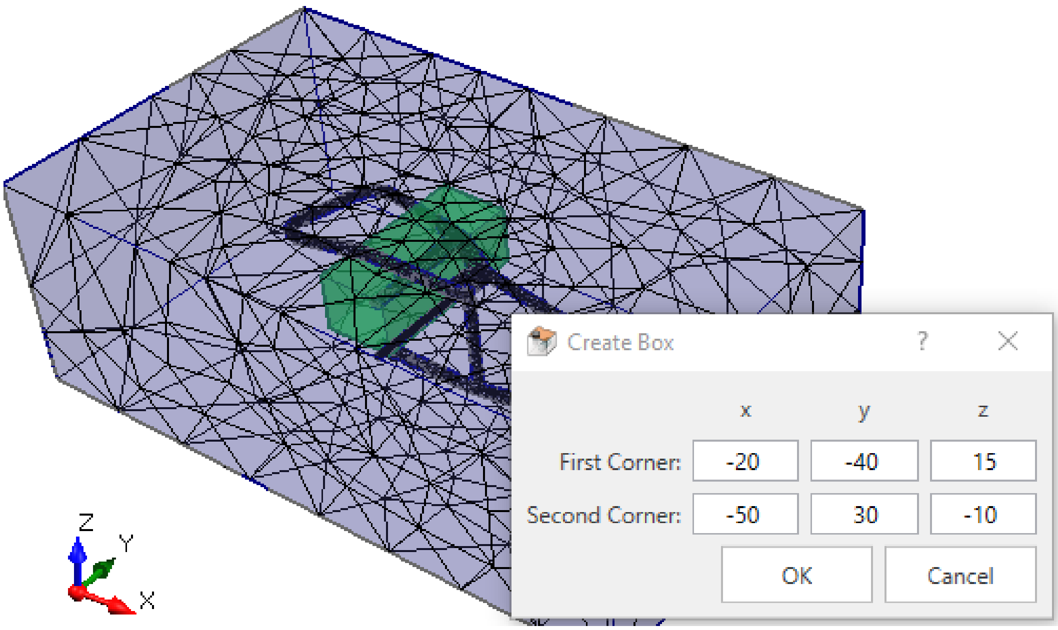
REFINING THE MESH

Refinement Regions

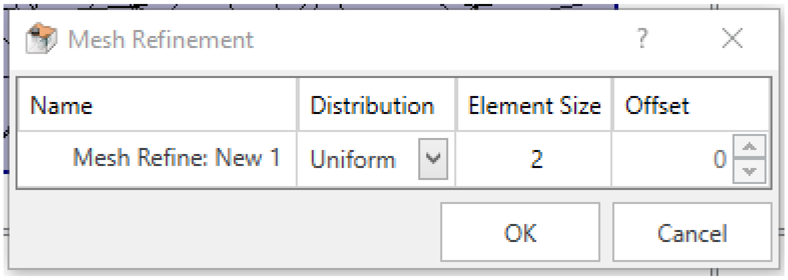
Next, we want to focus on one tunnel, so we will need to refine the mesh: Mesh → Define Refinement Regions.

The box that encapsulates the region of interest is: First Corner (x, y, z) = (-20, -40, 15), Second Corner = (-50, 30, -10), [OK].

Define Mesh Refinement Regions



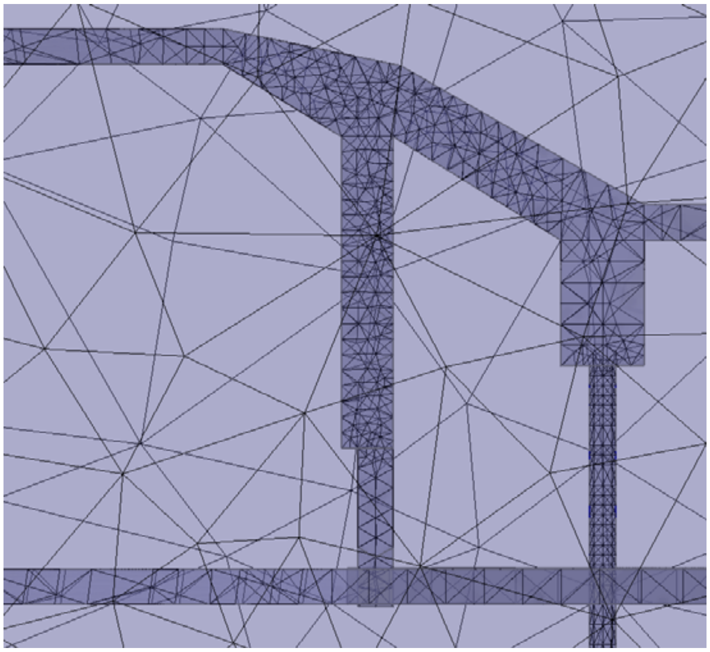
The green box will now be replaced by a red one, right-click in the viewport and press [Done]. Enter Element Size = 2 in the Mesh Refinement dialog, [OK].



Refine Graded Mesh

Select Mesh → Refine Graded Mesh. The area of interest now has a finer mesh.

Finer Mesh



Step 10: Computing Results

COMPUTE

Save

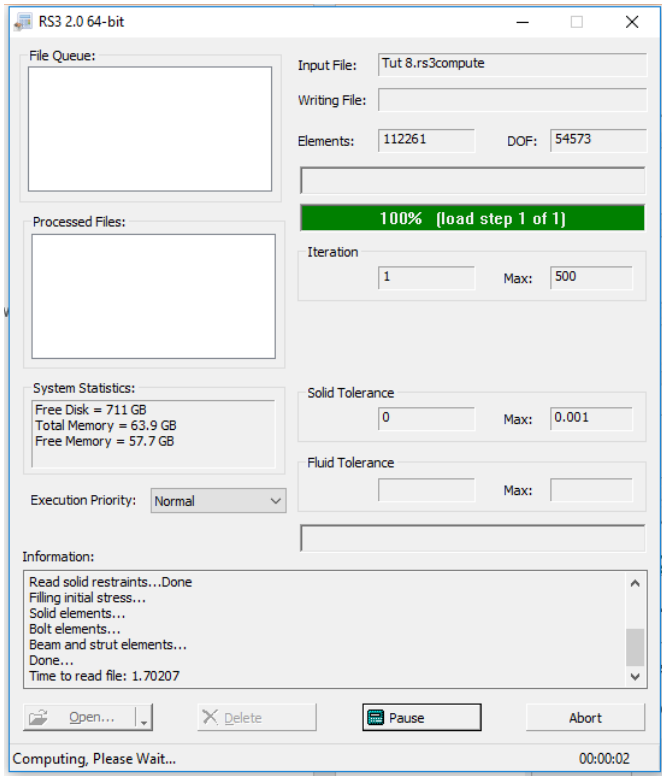
Next, move to the Compute tab. From this tab we can compute the results of our model. First, save: File → Save.

Use the Save As dialog to save the file. Next, you need to save the compute file: File → Save Compute File. You are now ready to compute the results.

Compute

Select: Compute → Compute.

Compute Engine



Step 11: Interpreting Results

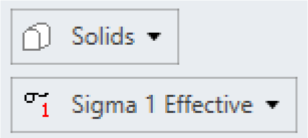
Results

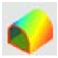
DISPLAYING THE RESULTS

Refresh Results

Next we move to the Results tab. From this tab we can analyze the results of our model. First, refresh the results: Interpret → Refresh Results.

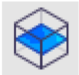
On the top right corner of the Results tab, you should see two drop down menus:





Excavation Contour

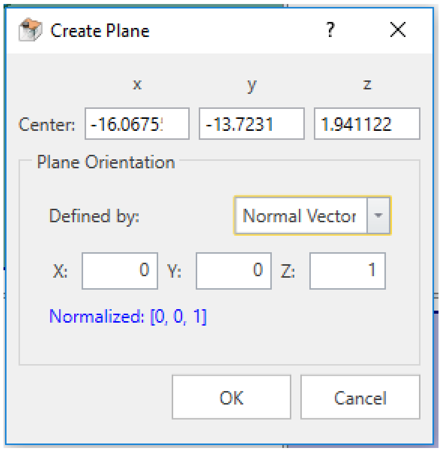
We will analyze a number of different "Data Type" results. Turn on the exterior contours such that we can see some results: Interpret → Show Excavation Contour.



XY Plane

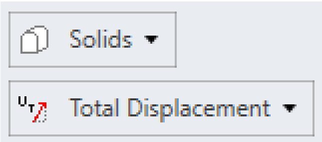
We also want to define a plane that goes through the slanted cross tunnel. First, we must define a plane, Interpret → XY Plane. In the Create Plane dialog, enter: Center (x, y, z) = (-16.0675, -13.7231, 1.941122), Normal (x, y, z) = (0, 0, 1), then press [OK].

Create Plane



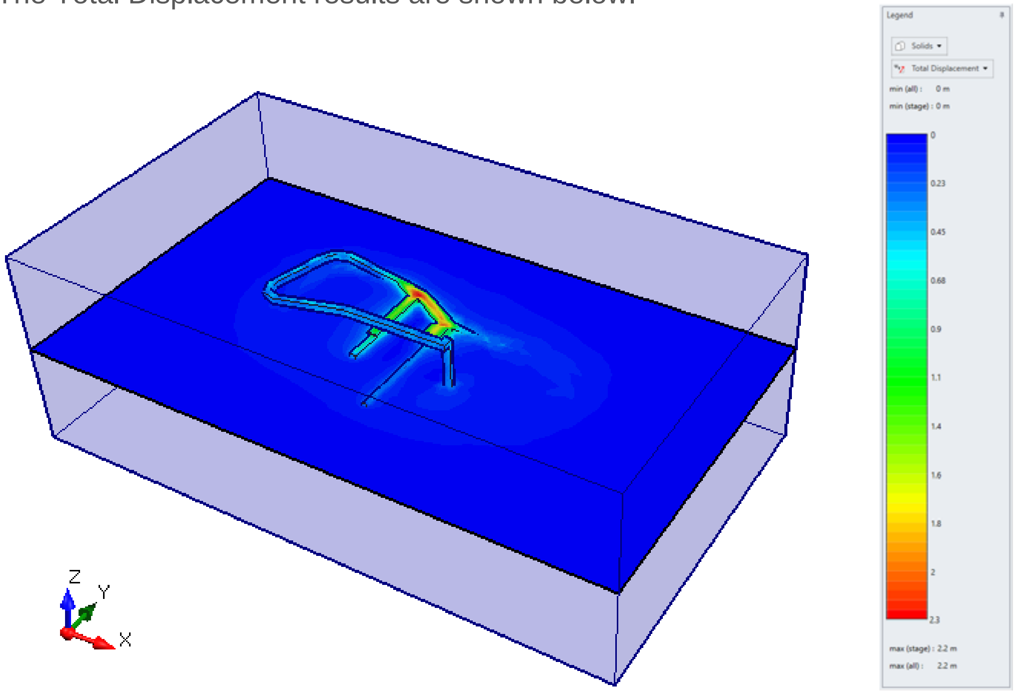
TOTAL DISPLACEMENT

In the top right corner of the Results tab, ensure Element = Solids, and change Data Type = Total Displacement:



The Total Displacement results are shown below.

Solids Total Displ



Other results are available to view as well. Thank you, this concludes the tutorial.